

SANDIA REPORT

SAND92—2292 • UC—705

Unlimited Release

Printed January 1993

Overview of the Sandia National Laboratories Engineering Analysis Code Access System

Gregory D. Sjaardema

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185 and Livermore, California 94550
for the United States Department of Energy
under Contract DE-AC04-76DP00789

Issued by Sandia National Laboratories, operated for the United States Department of Energy by Sandia Corporation.

NOTICE: This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government, any agency thereof or any of their contractors or subcontractors. The views and opinions expressed herein do not necessarily state or reflect those of the United States Government, any agency thereof or any of their contractors.

Printed in the United States of America. This report has been reproduced directly from the best available copy.

Available to DOE and DOE contractors from
Office of Scientific and Technical Information
PO Box 62
Oak Ridge, TN 37831

Prices available from (615) 576-8401, FTS 626-8401

Available to the public from
National Technical Information Service
US Department of Commerce
5285 Port Royal Rd
Springfield, VA 22161

NTIS price codes
Printed copy: A03
Microfiche copy: A01

SAND92-2292
Unlimited Release
Printed January 1993

Distribution
Category UC-705

Overview of the Sandia National Laboratories Engineering Analysis Code Access System

Gregory D. Sjaardema
Solid and Structural Mechanics Department
Sandia National Laboratories
Albuquerque, NM 87185

Abstract

The Sandia National Laboratories (SNL) Engineering Analysis Code Access System (SEACAS) is a collection of structural and thermal codes and utilities used by analysts at SNL. The system includes pre- and post-processing codes, analysis codes, database translation codes, support libraries, UNIX^{*} shell scripts, and an installation system.

SEACAS is used at SNL on a daily basis as a production, research, and development system for the engineering analysts and code developers. Over the past year, approximately 180 days of Cray Y-MP[†] CPU time have been used at SNL by SEACAS codes. The job mix includes jobs using only a few seconds of CPU time, up to jobs using two and one-half days of CPU time. SEACAS is running on several different systems at SNL including Cray Unicos, Hewlett Packard HP-UX[‡], Digital Equipment Ultrix[§], and Sun SunOS^{**}.

This document is a short description of the codes the SEACAS system.

*UNIX is a registered trademark of UNIX Systems Laboratories Inc.

†Cray and Unicos are registered trademarks of Cray Research, Inc.

‡HP-UX is a registered trademark of Hewlett-Packard Company.

§Ultrix is a registered trademark of Digital Equipment Corporation.

**Sun and SunOS are registered trademarks of Sun Microsystems, Inc.

Table of Contents

Philosophy.....	5
SEACAS Code Types.....	6
Preprocessing and Postprocessing Codes	6
Analysis Programs	6
Translation Codes	7
Support Libraries	7
Scripts	7
Constitutive Models	8
SEACAS Abstracts and References	8
Algebra	8
Blot	8
Conchas	9
Exodus and Genesis File Format	9
Fastq	9
Grobe	10
Numbers	10
Pronto2D and Pronto3D	10
Subway3D	12
Sancho	12
Santos and Santos3D	13
Jac2D, Jac3D, and Jacq3D	13
Merlin II	14
Supes Library	14
Gen3D	14
Gjoin	15
Grepos	15
Aprepro	15
Constitutive Models	16
References.....	16

List of Figures

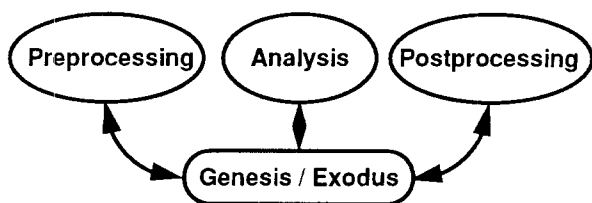
Modular Structure of SEACAS	5
Expanded Modular View of SEACAS	5
Structure of Mesh Generation Process	5

Philosophy

SEACAS is a modular system based upon a common binary datafile format called EXODUS¹ which includes the mesh description and the timeplanes of the computed results. A subset of this format, called GENESIS², is used to refer to the mesh description portion of the EXODUS format.

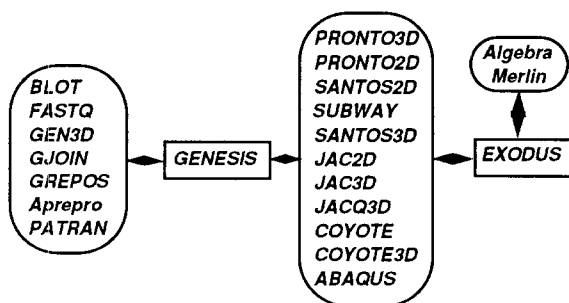
All of the preprocessing, analysis, postprocessing, and translation codes can read and/or write EXODUS database files. A schematic of this is shown in Figure 1.

FIGURE 1. Modular Structure of SEACAS



With this structure, codes can be tailored for a single function. For example, an analysis code can be added to the system without writing new mesh generation and post processing programs. Also, code to code data transfers and restarts use the EXODUS format. A more complete view of this is shown in Figure 2.

FIGURE 2. Expanded Modular View of SEACAS

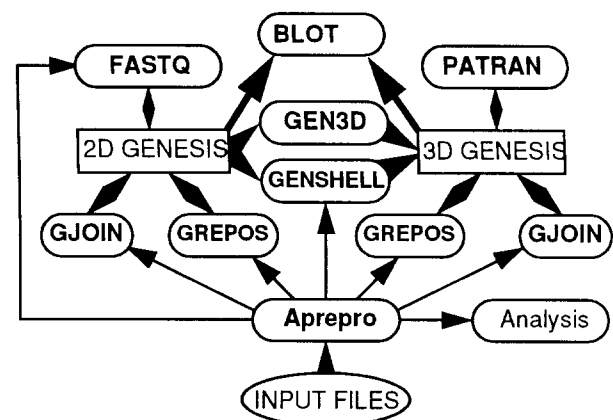


This modular concept is used extensively in the mesh generation process. Several special-purpose codes have been written that each perform a certain mesh generation task. Examples of this are:

- generate two-dimensional mesh (fastq).
- transform two-dimensional mesh into a three-dimensional mesh (gen3d, genshell).
- join two or more 2D or 3D meshes into a single mesh (gjoin).
- reposition a 2D or 3D mesh (grepos).

This structure is shown in Figure 3. Finite element meshes of complicated three-dimensional geometries have been successfully generated using this system which is built up of codes that singly provide a limited capability, but when used as a system are very powerful.

FIGURE 3. Structure of Mesh Generation Process



All of the codes are written in as portable a form as possible. Fortran codes are written in ANSI Standard FORTRAN-77³ and C language codes are written in ANSI Standard C⁴ where possible. Machine-specific routines are limited in number and are grouped together to minimize the time required to adapt them to a new system. SEACAS codes have been ported to several Unix systems including Cray, HP, Sun, Dec, Stardent, Convex, and SGI.

A code management system called **cvs**⁵ is used for all of the SEACAS codes to provide traceability and retrievability for Quality Assurance. The change logs include who changed the code, when it was changed, what was changed, and why the change was made. If required, any previous version can be retrieved.

SEACAS Code Types

SEACAS is divided into five broad categories of codes: prepost, analysis, translators, libraries, and scripts. The categories can be roughly defined as follows:

prepost: Pre- and Postprocessing codes including mesh generation, visualization, preprocessors, and database manipulation codes.

analysis: Finite element analysis codes including quasistatic, transient dynamics, thermal, and electromechanics. Two- and three-dimensional, nonlinear, large deformation.

translators: Translation codes for editing output files (EXODUS), inter-machine translation, and exodus from/to commercial database translation

libraries: Support libraries including database routines, common machine-specific routines, plot routines, graphics device drivers, and interactive help routines.

scripts: Unix shell scripts for executing the prepost, analysis, and translation codes. Also includes support and installation routines.

Preprocessing and Postprocessing Codes

The pre- and postprocessing codes are comprised of the following mesh generation, visualization, and database manipulation codes.

gjoin: Join together two or more GENESIS databases into a single GENESIS database.

gen3d: Transform a two-dimensional GENESIS database into a three-dimensional GENESIS database. Several transformations are supported and additional transformations can be easily added.

grepos: Reposition or scale a GENESIS database.

fastq: Interactive two-dimensional finite element mesh generation program. Includes several mesh generation options including paving.

aprepro: An algebraic preprocessing program.

genshell: Transform a two-dimensional GENESIS database into a three-dimensional shell GENESIS database. Several transformations are supported and additional transformations can be easily added.

numbers: Calculates several properties of an EXODUS file, including mass properties, timesteps, condition numbers, cavity volumes, and others.

grope: Interactively examine an EXODUS database. Grope is also contained within the blot program. Grope is primarily used to validate EXODUS files.

blot: The primary graphical two-dimensional and three-dimensional postprocessing code. It includes deformed mesh plots, contour plots, shaded fringe plots, variable versus variable and time history plots, and distance versus variable plots.

Analysis Programs

The analysis codes include quasistatic, transient dynamics, thermal, and electromechanics codes.

jac2d: Jac2d is a finite element program which uses a nonlinear conjugate gradient technique to solve the large displacement, large strain, temperature dependent and material nonlinear problem for two-dimensional plane or axisymmetric solids.

jac3d: Jac3d is a finite element program which uses a nonlinear conjugate gradient technique to solve the large displacement, large strain, temperature dependent and material nonlinear problem for three dimensional solids.

jacq3d: Jacq3d is a finite element program which uses a nonlinear conjugate gradient technique to solve the thermal conduction problem for three dimensional solids.

pronto2d: Pronto2d is a two-dimensional (planar and axisymmetric) transient solid dynamics code. Lagrangian formulation with explicit time integra-

tion is used for analyzing large deformations of highly nonlinear materials subjected to extremely high strain rates.

pronto3d: Pronto3d is a three-dimensional transient solid dynamics code. Lagrangian formulation with explicit time integration is used for analyzing large deformations of highly nonlinear materials subjected to extremely high strain rates.

sancho: Sancho is a finite element program which uses a dynamic relaxation technique to compute the quasistatic, large deformation, temperature dependent, inelastic response of planar or axisymmetric solids.

santos: Santos is a two-dimensional finite element program which uses a dynamic relaxation technique to compute the quasistatic, large deformation, temperature dependent, inelastic response of planar or axisymmetric solids. Newer than sancho.

santos3d: Santos3d is a three-dimensional version of santos. It is currently in development.

conchas: Conchas is a linear structural analysis code for axisymmetric structures with loads that are symmetric about a plane.

subway: Subway is a three-dimensional finite element program for calculating the transient electro-mechanical response of dielectric materials.

Translation Codes

The translation codes are used to convert EXODUS databases into different formats and to edit EXODUS databases.

algebra: Manipulates EXODUS finite element output data by evaluating algebraic expressions. Equation variables are dependent on the input database variable names.

seaexo: Converts SEACO files (the binary file format the preceded EXODUS) into EXODUS files.

exotxt and **txtexo:** Converts EXODUS to and from a specially formatted ASCII text file.

exoexo: A program which simply translates an EXODUS file to the same EXODUS file. It is used as a base program for writing new translators or database manipulation programs.

abaexo: Converts ABAQUS (commercial finite element code) results files to EXODUS

conex: Concatenates several EXODUS files into a single EXODUS file. Used to create a single EXODUS file from analyses that have been restarted.

exoxdr and **xdrexo:** Converts EXODUS to and from an external data representation format (XDR) which can be transmitted between computers of different architectures, word lengths, and byte orders.

exogen: Creates a GENESIS mesh database from a specific time step of an EXODUS file. Used when an analysis of a deformed geometry is required. For example, an impact analysis followed by a thermal analysis.

merlin2: Transfers (maps) nodal data between finite element meshes. For example, thermal output from a heat conduction code to a thermal input file for a quasistatic mechanics code.

exopat and **patexo:** Converts EXODUS to and from PATRAN (commercial mesh generation program) neutral file format.

In addition to the translation codes listed above, the Cray Unicos system provides the capability to translate EXODUS files into a VMS and IEEE format using the exoexo code.

Support Libraries

The support libraries include routines which are used in several of the codes.

supes: A library of frequently used non-numerical routines for input command parsing, dynamic memory management, and system dependent utilities.

suplib: A library of routines for reading and writing EXODUS files, manipulating string, input parsing, and other tasks.

xhelp: An interactive and code-callable help system.

plt: A medium-level plot library used by fastq, blot, graph, and post.

svdi: A low-level device-interface library.

Scripts

The scripts category is composed of the Unix shell scripts for executing the preprocessing, postprocessing, analysis, and translation codes; and the imake files required to install SEACAS.

Constitutive Models

Many constitutive models are implemented in the analysis codes.

Elastic: Typical linear elastic material using Hooke's Law.

Elastic-Plastic with Combined Hardening: Standard von Mises type yield condition with combined kinematic and isotropic linear hardening.

Viscoplastic: Simple rate-dependent plasticity.

Viscoelastic: Thermoviscoelastic model for glass solidification

Damage: Dynamic fracture of brittle rock.

Soils and Crushable Foam: Volumetric plasticity model.

Low Density Foam: Low density polyurethane foam behavior.

Hydrodynamic: Used with Equations of State in PRONTO

Elastic-Plastic Hydrodynamic: Combination of elastic-plastic combined hardening with hydrodynamic pressure response.

Rate and Temperature Dependent Plasticity: Unified creep plasticity.

Secondary Creep: Power hardening steady state creep with elastic bulk response.

Isotropic Elastic/Plastic Power-Law Hardening with Luders Strain.

Johnson-Cook Strength: Rate and Temperature-dependent plasticity.

Hyperelastic: Stress based on the principal strains.

Salt Consolidation: Power hardening steady state creep deviatoric response with consolidation bulk response.

Not all of the constitutive models are implemented in all of the analysis codes. The quasistatic analysis codes typically implement the constitutive models using temperature-dependent properties to facilitate thermal-stress calculations.

SEACAS Abstracts and References

Abstracts and References for selected analysis, preprocessing, postprocessing, translation, and library codes are listed in this section. The codes not listed are primarily translators which are not formally documented.

Algebra

The Algebra program allows the user to manipulate data from a finite element analysis before it is plotted. The finite element output data is in the form of variable values (e.g., stress, strain, and velocity components) in an EXODUS database. The Algebra program evaluates user-supplied functions of the data and writes the results to an output EXODUS database which can be read by plot programs.

A. P. Gilkey, "ALGEBRA - A Program that Algebraically Manipulates the Output of a Finite Element Analysis (EXODUS Version)," SAND88-1431, Sandia National Laboratories, Albuquerque, New Mexico, August 1988.

Blot

Blot is a graphics program for post-processing of finite element analyses output in the EXODUS database format. It is command driven with free-format input and can drive any graphics device supported by the Sandia Virtual Device Interface.

Blot produces mesh plots with various representations of the analysis output variables. The major mesh plot capabilities are deformed mesh plots, line contours, filled (painted) contours, vector plots of two/three variables (e.g., velocity vectors), and symbol plots of scalar variables (e.g., discrete cracks). Pathlines of analysis variables can also be drawn on the mesh. Blot's features include element selection by

material, element birth and death, multiple views for combining several displays on each plot, symmetry mirroring, and node and element numbering.

Blot can also produce X-Y curve plots of the analysis variables. Blot generates time-versus-variable plots or variable-versus-variable plots. It also generates distance-versus-variable plots at selected time steps where the distance is the accumulated distance between pairs of nodes or element centers.

Amy P. Gilkey and John H. Glick, "BLOT-A Mesh and Curve Plot Program for the Output of a Finite Element Analysis," SAND88-1432, Sandia National Laboratories, Albuquerque, New Mexico, August 1991.

R. J. Meyers, "Updates to the postprocessing program BLOT," memo to distribution dated August 21, 1990, Sandia National Laboratories, Albuquerque, NM.

Conchas

CONCHAS is a linear finite element structural analysis code which is specialized for axisymmetric structures. Loads and responses are limited to those symmetric about a plane which includes the symmetric axis of the structure. CONCHAS will perform eigenanalysis, static analysis, and dynamic analysis. The element library includes consistent-mass shell, solid, and beam elements, nonlinear springs, and concentrated masses. Pre- and postprocessing is available with the separately supported utilities BLOT, PATRAN, and FASTQ.

William C. Mills-Curran and Dennis P. Flanagan, "CONCHAS Users Manual," SAND88-1006, Sandia National Laboratories, Albuquerque, New Mexico, June 1989.

Exodus and Genesis File Format

William. C. Mills-Curran, Amy P. Gilkey, and Dennis P. Flanagan, "EXODUS: A Finite Element File Format for Pre- and Post-processing," SAND87-2977, Sandia National Laboratories, Albuquerque, New Mexico, September 1988.

L. M. Taylor and D. P. Flanagan and W. C. Mills-Curran, "The GENESIS Finite Element Mesh File Format," SAND86-0910, Sandia National Laboratories, Albuquerque, NM, May 1986.

Fastq

The FASTQ code is an interactive two-dimensional finite element mesh generation program. It is designed to provide a powerful and efficient tool to both reduce the time required of an analyst to generate a mesh, and to improve the capacity to generate good meshes in arbitrary geometries. It is based on a mapping technique and employs a set of higher-order primitives which have been developed for automatic meshing of commonly encountered shapes (i.e. the triangle, semi-circle, etc.) and conditions (i.e. mesh transitioning from coarse to fine mesh size). FASTQ has been designed to allow user flexibility and control. The user interface is built on a layered command level structure. Multiple utilities are provided for input, manipulation, and display of the geometric information, as well as for direct control, adjustment, and display of the generated mesh. Enhanced boundary flagging has been incorporated and multiple element types and output formats are supported. FASTQ includes the paving algorithm which meshes arbitrary two-dimensional geometries with an all quadrilateral mesh.

T. D. Blacker, "FASTQ Users Manual, Version 2.1," SAND88-1326, Sandia National Laboratories, Albuquerque, NM, July 1988.

This paper presents a new mesh generation technique, paving, which meshes arbitrary two-dimensional geometries with an all-quadrilateral mesh. Paving allows varying element size distributions on the boundary as well as the interior of a region. The generated mesh is well formed (i.e. nearly square elements, elements perpendicular to boundaries, etc.) and geometrically pleasing (i.e. mesh contours tend to follow geometric contours of the boundary). In this paper we describe the theory behind this algorithmic/heuristic technique, evaluate the performance of the approach and present examples of automatically generated meshes.

T. D. Blacker and M. B. Stephenson, "Paving: A New Approach to Automated Quadrilateral Mesh Generation," *International Journal for Numerical Methods in Engineering*, Volume 32, pp. 811-847, 1991.

Grope

Grope is a program that examines the input to a finite element analysis (which is in the GENESIS database format) or the output from an analysis (in the EXODUS database format). Grope allows the user to examine any value in the database. The display can be directed to the user's terminal or to a print file.

G. D. Sjaardema, "GROPE - A GENESIS/EXODUS Database Examination Program," SAND92-2289, Sandia National Laboratories, Albuquerque, NM, October 1992

Numbers

Numbers is a program which reads and stores data from a finite element model described in the EXODUS database format. Within this program are several utility routines which generate information about the finite element model. The utilities currently implemented in Numbers allow the analyst to determine information such as

(1) the volume and coordinate limits of each of the materials in the model; (2) the mass properties of the model; (3) the minimum, maximum, and average element volumes for each material; (4) the volume and change in volume of a cavity; (5) the nodes or elements that are within a specified distance from a user-defined point, line, or plane; (6) an estimate of the explicit central-difference timestep for each material; (7) the validity of contact surfaces or slidelines, that is, whether two surfaces overlap at any point; and (8) the distance between two surfaces.

G. D. Sjaardema, "NUMBERS: A Collection of Utilities for Pre- and Postprocessing Two- and Three-Dimensional EXODUS Finite Element Models," SAND88-0737, Sandia National Laboratories, Albuquerque, New Mexico, March 1989.

Pronto2D and Pronto3D

PRONTO 2D is a two-dimensional transient solid dynamics code for analyzing large deformations of highly nonlinear materials subjected to extremely high strain rates. This Lagrangian finite element program uses an explicit time integration operator to integrate the equations of motion. Four node uniform strain quadrilateral elements are used in the finite element formulation. A number of new numerical algorithms which have been developed for the code are described in this report. An adaptive time step control algorithm is described which greatly improves stability as well as performance in plasticity problems. A robust hourglass control scheme which eliminates hourglass distortions without disturbing the finite element solution is included. All constitutive models in PRONTO are cast in an unrotated configuration defined using the rotation determined from the polar decomposition of the deformation gradient. An accurate incremental algorithm was developed to determine this rotation and is described in detail. A robust contact algorithm was developed which allows for the impact and interaction of deforming contact surfaces of quite general geometry.

A number of numerical examples are presented to demonstrate the utility of these algorithms.

Lee M. Taylor and Dennis P. Flanagan, "PRONTO 2D, A Two-Dimensional Transient Solid Dynamics Program," SAND86-0594, Sandia National Laboratories, Albuquerque, New Mexico, March 1987.

PRONTO 3D is a three-dimensional transient solid dynamics code for analyzing large deformations of highly nonlinear materials subjected to extremely high strain rates. This Lagrangian finite element program uses an explicit time integration operator to integrate the equations of motion. Eight-node uniform strain hexahedral elements are used in the finite element formulation. A number of new numerical algorithms which have been developed for the code are described in this report. An adaptive time step control algorithm is described which greatly improves stability as well as performance in plasticity problems. A robust hourglass control scheme which eliminates hourglass distortions without disturbing the finite element solution is included. All constitutive models in PRONTO are cast in an unrotated configuration defined using the rotation determined from the polar decomposition of the deformation gradient. An accurate incremental algorithm was developed to determine this rotation and is described in detail. A robust contact algorithm was developed which allows for the impact and interaction of deforming contact surfaces of quite general geometry. Numerical examples are presented to demonstrate the utility of these algorithms.

L. M. Taylor and D. P. Flanagan, "PRONTO 3D, A Three-Dimensional Transient Solid Dynamics Program," SAND87-1912, Sandia National Laboratories, Albuquerque, New Mexico, March, 1989.

An external code interface is defined which allows other transient applications to communicate with the PRONTO family of finite element programs. This interface is

written in ANSI FORTRAN and allows an independent author to specify requirements for an external code to PRONTO. The interface is written such that updates to PRONTO will not require modifications to the external code.

L. M. Taylor and D. P. Flanagan, "An External Code Interface for the PRONTO Family of Transient Solid Dynamics Programs," SAND87-3003, Sandia National Laboratories, Albuquerque, New Mexico, September, 1988.

PRONTO 2D and PRONTO 3D are two- and three-dimensional solid dynamics codes for analyzing large deformations of highly nonlinear materials subjected to high strain rates. This newsletter is issued to document changes to these codes. As of this writing, the latest version of PRONTO 2D is Version 4.5.6, and the latest version of PRONTO 3D is Version 4.5.6.

This update of the two codes discusses two major modifications to the numerical formulations, three new constitutive models, and the additions and improvements of contact surfaces. Changes in file formats, other miscellaneous revisions, and the availability of PRONTO 2D and PRONTO 3D are also discussed. In addition, updated commands for PRONTO 2D are provided in Appendix A of this newsletter.

S. W. Attaway, "Update of PRONTO 2D and PRONTO 3D Transient Solid Dynamics Program," SAND90-0102, Sandia National Laboratories, Albuquerque, New Mexico, November, 1990.

PRONTO 3D is a three-dimensional transient solid dynamics code for analyzing large deformations of highly nonlinear materials subjected to high strain rates. It is a Lagrangian finite element program with explicit integration of the equations of motion through time. This report documents the implementation of a four-node quadrilateral shell element into Version 6.0 of PRONTO 3D.

This report describes the theory, implementation and use of a four-node shell

element. Also described are the required architectural changes made to PRONTO 3D to allow multiple element types. Several test problems are documented for verification of the PRONTO 3D implementation and for demonstration of computational savings using shell elements for thin structures. These problems also serve as examples for the user. A complete, updated list of the PRONTO 3D input commands is also included.

V. L. Bergmann, "Transient Dynamics Analysis of Plates and Shells with PRONTO 3D," SAND91-1182, Sandia National Laboratories, Albuquerque, New Mexico, September 1991.

This update discusses modifications of PRONTO 3D tailored to the design of fast burst nuclear reactors. A thermoelastic constitutive model and spatially variant thermal history load were added for this special application. Included are descriptions of the thermoelastic constitutive model and the thermal loading algorithm, two example problems used to benchmark the new capability, a user's guide, and PRONTO 3D input files for the example programs. The results from PRONTO 3D thermoelastic finite element analysis are benchmarked against measured data and finite difference calculations.

PRONTO 3D is a three-dimensional transient solid dynamics code for analyzing large deformations of highly non-linear materials subjected to high strain rates. The code modifications are implemented in PRONTO 3D Version 5.3.3.

D. S. Oscar, S. W. Attaway, and J. D. Miller, "Modifications of the PRONTO 3D Finite Element Program Tailored to Fast Burst Nuclear Reactor Design," SAND91-0959, Sandia National Laboratories, Albuquerque, New Mexico, August 1991.

Subway3D

SUBWAY 3D is a three-dimensional finite element code for numerical simulation of the

transient electromechanical response of dielectric materials. The code uses a preconditioned conjugate gradient method to solve Poisson's equation governing the electric potential in the dielectrics. This field solver is embedded in a modified version of the finite element transient dynamics code PRONTO 3D and allows solution for the electric response at each time step used in the explicit integration of the equations of motion. The code is structured to allow flexibility in formulation of initial-boundary value problems by permitting specification of electrical conductors and allowing these conductors to be connected to electrical circuits isolated from the mechanical deformations. An algorithm for solution of these initial-boundary value problems is incorporated into the code with special material models to represent the response of ordinary dielectrics and electromechanically active dielectrics like piezoelectric and ferroelectric ceramics. The code has a wide range of applications and, in particular, can be used to calculate responses of shock activated power supplies, impact gages, and the change in electrical capacitance due to deformations.

S. T. Montgomery, documentation not yet available

Sancho

SANCHO is a finite element computer program designed to compute the quasistatic, large deformation, inelastic response of planar or axisymmetric solids. Finite strain constitutive theories for plasticity, volumetric plasticity, and metallic creep behavior are included. A constant bulk strain, bilinear displacement isoparametric finite element is employed for the spatial discretization. The solution strategy used to generate the sequence of equilibrium solutions is a self-adaptive dynamic relaxation scheme which is based on explicit central difference pseudo-time integration and artificial damping. A master-slave algorithm for sliding interfaces is also implemented. A theoretical development of the appropriate governing equations and a

description of the numerical algorithms are presented along with a user's guide which includes several sample problems and their solution.

Charles M. Stone and Raymond D. Krieg and Zelma E. Beisinger, "SANCHO, A Finite Element Computer Program for the Quasistatic, Large Deformation, Inelastic Response of Two-Dimensional Solids," SAND84-2618, Sandia National Laboratories, Albuquerque, New Mexico, April 1985.

Santos and Santos3D

SANTOS is a finite element computer program designed to compute the quasistatic, large deformation, inelastic response of planar or axisymmetric solids. SANTOS is based on the dynamics program PRONTO2D by L. M. Taylor and D. P. Flanagan. SANTOS utilizes a self-adaptive dynamic relaxation algorithm to achieve a quasistatic solution. The efficiency, speed, through vectorization, and state-of-the-art algorithms that Taylor and Flanagan built into PRONTO2D are maintained in SANTOS. The architecture of the code as well as the user interface is similar for both codes which improves code reliability and encourages the use of both codes by analysts. By utilizing the same material interface, the same constitutive models may be utilized by both codes which allows for coupling of the two codes.

C. M. Stone, documentation not yet available.

Jac2D, Jac3D, and Jacq3D

The nonlinear conjugate gradient procedure is employed in the computer program JAC2D to solve quasi-static nonlinear mechanics problems. A set of continuum equation which describes accurately nonlinear mechanics involving large rotation and strain is very conveniently used with the conjugate gradient method to solve the nonlinear problem. The method is exploited in a two-dimensional setting while using

various methods for accelerating convergence. Sliding interface condition are also implemented. A four-node Lagrangian uniform strain element is used with hourglass stiffness to control the zero energy nodes. Materials which can be modeled are elastic and isothermal elastic-plastic with combined kinematic and isotropic hardening. The program is vectorized to perform very efficiently on CRAY computers. Sample problems described are the bending of a thin beam, the rotation of a unit cube, a pressurized and thermally loaded sphere and cylinder.

J. H. Biffle, "A Two-Dimensional Finite Element Computer Program for the Non-Linear Quasistatic Response of Solids with the Conjugate Gradient Method," SAND81-0998, Sandia National Laboratories, Albuquerque, New Mexico, April 1984.

JAC3D is a three-dimensional finite element program designed to solve quasi-static nonlinear mechanics problems. A set of continuum equations, which describe nonlinear mechanics involving large rotation and strain, is used. A nonlinear conjugate gradient method is used to solve the nonlinear equations. The method is implemented in a three-dimensional setting with various methods for accelerating convergence. Sliding interface conditions are also implemented. An eight-node Lagrangian uniform strain element is used with hourglass stiffness to control the zero energy modes. This release of the code contains elastic and isothermal elastic-plastic material models. Other material models can be added relatively easily. The program is vectorized to perform very efficiently on CRAY computers. Sample problems described are the bending of a thin beam, the rotation of a unit cube, a pressurized and thermally loaded sphere.

J. H. Biffle, "JAC3D - A Three-Dimensional Finite Element Computer Program for the Non-Linear Quasistatic Response of Solids with the Conjugate Gradient Method," In preparation.

The nonlinear conjugate gradient procedure is employed in the computer program JACQ3D to solve the steady state and transient nonlinear heat conduction problem for solids in three dimensions. The program can also be used for other types of diffusion problems. The problem is formulated with the finite element method and employs an eight node uniform gradient element with hourglass stiffness to control the zero energy modes. JACQ3D is highly vectorized to perform efficiently on CRAY computer. Sample problems are included to verify the code and to provide examples of the use of the code.

J. H. Biffle, "JACQ3D - A Three-Dimensional Finite Element Computer Program for Non-Linear Heat Conduction Problems with the Conjugate Gradient Method," In preparation.

Merlin II

The MERLIN II program is designed to transfer data between finite element meshes of arbitrary geometry. The program is structured to accurately interpolate previously computed solutions onto a given mesh and format the resulting data for immediate use in another analysis program. Data from either two-dimensional or three-dimensional meshes may be considered. The theoretical basis and computational algorithms used in the program are described and complete user instructions are presented. Several example problems are included to demonstrate program usage.

D. K. Gartling, "MERLIN II - A Computer Program to Transfer Solution Data Between Finite Element Meshes," SAND89-2989, Sandia National Laboratories, Albuquerque, New Mexico, July 1991.

Supes Library

The Software Utilities Package for the Engineering Sciences (SUPES) is a collection of subprograms which perform

frequently used non-numerical services for the engineering applications programmer. The three functional categories of SUPES are: (1) input command parsing, (2) dynamic memory management, and (3) system dependent utilities. The subprograms in categories one and two are written in standard FORTRAN-77, while the subprograms in category three are written to provide a standardized FORTRAN interface to several system dependent features.

J. H. Red-Horse, W. C. Mills-Curran, D. P. Flanagan, "SUPES Version 2.1, A Software Utilities Package for the Engineering Sciences," SAND90-0247, Sandia National Laboratories, Albuquerque, New Mexico, May 1990.

Gen3D

GEN3D is a three-dimensional mesh generation program. The three-dimensional mesh is generated by mapping a two-dimensional mesh into three-dimensions according to one of four types of transformations: translating, rotating, mapping onto a spherical surface, and mapping onto a cylindrical surface. The generated three-dimensional mesh can then be reoriented by offsetting, reflecting about an axis, and revolving about an axis. GEN3D can be used to mesh geometries that are axisymmetric or planar, but, due to three-dimensional loading or boundary conditions, require a three-dimensional finite element mesh and analysis. More importantly, it can be used to mesh complex three-dimensional geometries composed of several sections when the sections can be defined in terms of transformations of two-dimensional geometries. The code GJOIN is then used to join the separate sections into a single body. GEN3D reads and writes two-dimensional and three-dimensional mesh databases in the GENESIS database format; therefore, it is compatible with the preprocessing, postprocessing, and analysis codes used by the Engineering Analysis Department at Sandia National Laboratories, Albuquerque, NM.

Amy P. Gilkey and Gregory D. Sjaardema, "GEN3D: A GENESIS Database 2D to 3D Transformation Program," SAND89-0485, Sandia National Laboratories, Albuquerque, New Mexico, March 1989.

This memo describes the changes that have been made to the GEN3D program since the manual (SAND89-0485) was published. The changes include: (1) New mesh generation options: spline, project, twist, interval, and rotcen transformations; (2) New mesh modification options: change material, change sideset, and change nodeset; (3) New mesh orientation option: scale; and (4) Miscellaneous changes.

G. D. Sjaardema, "Updates to the mesh generation program GEN3D," memo to distribution dated April 11, 1990, Sandia National Laboratories, Albuquerque, New Mexico.

Gjoin

GJOIN is a two- or three-dimensional mesh combination program. GJOIN combines two or more meshes written in the GENESIS mesh database format into a single GENESIS mesh. Selected nodes in the two meshes that are closer than a specified distance can be combined. The geometry of the mesh databases can be modified by scaling, offsetting, revolving, and mirroring. The combined meshes can be further modified by deleting, renaming, or combining material blocks, sideset identifications, or nodeset identifications. GJOIN is one of the mesh generation tools in the Sandia National Laboratories Engineering Analysis Code Access System (SEACAS). GJOIN is typically used with the other SEACAS mesh generation codes GEN3D, GENSHLL, GREPOS, and Aprepro.

G. D. Sjaardema, "GJOIN: A Program for Merging Two or More GENESIS Databases," (memo to distribution dated 19 June 1991), Sandia National Laboratories, Albuquerque, New Mexico.

Grepos

Grepos is a mesh utility program that repositions or modifies the configuration of a 2D or 3D mesh. Grepos can be used to change the orientation and size of a 2D or 3D mesh; change the material block, nodeset, and sideset IDs; or "explode" the mesh to facilitate viewing of the various parts of the model. Grepos also updates the EXODUS QA and information records to help track the codes and files used to generate the mesh. GREPOS reads and writes 2D and 3D mesh databases in the GENESIS database format; therefore, it is compatible with the preprocessing, postprocessing, and analysis codes in SEACAS.

G. D. Sjaardema, "GREPOS: A GENESIS Database Repositioning Program," SAND90-0566, Sandia National Laboratories, Albuquerque, New Mexico, April 1990.

Aprepro

Aprepro is an algebraic preprocessor that reads a file containing both general text and algebraic, string, or conditional expressions. It interprets the expressions and outputs them to the output file along with the general text. The syntax used in Aprepro is such that all expressions between the delimiters { and } are evaluated and all other text is simply echoed to the output file. Aprepro contains several mathematical functions, string functions, and flow control constructs. In addition, functions are included that, with some additional files, implement a units conversion system and a material database lookup system. Aprepro was written primarily to simplify the preparation of parameterized input files for finite element analyses at Sandia National Laboratories; however, it can process any text file that does not use the characters { }.

References

G. D. Sjaardema, "Aprepro: An Algebraic Pre-processor for Parameterizing Finite Element Analyses," SAND92-2291, Sandia National Laboratories, Albuquerque, New Mexico, December 1992.

Constitutive Models

B. J. Thorne, "A Damage Model for Rock Fragmentation and Comparison of Calculations With Blasting Experiments in Granite," SAND90-1389, Sandia National Laboratories, Albuquerque, New Mexico, October 1990.

E. P. Chen, "A Computational Model for Jointed Media with Orthogonal Sets of Joints," SAND86-1122, Sandia National Laboratories, Albuquerque, New Mexico, March 1987.

M. K. Neilsen, H. S. Morgan, R. D. Krieg, "A Phenomenological Constitutive Model for Low Density Polyurethane Foams," SAND86-2927, Sandia National Laboratories, Albuquerque, New Mexico, April 1987.

R. S. Chambers, "A Viscoelastic Material Model for Computing Stresses in Glass," SAND90-0645, Sandia National Laboratories, Albuquerque, New Mexico, July 1990.

G. D. Sjaardema, R. D. Krieg, "A Constitutive Model for the Consolidation of WIPP Crushed Salt and Its Use in Analyses of Backfilled Shaft and Drift Configurations," SAND87-1977, Sandia National Laboratories, Albuquerque, New Mexico, October 1987.

C. M. Stone, G. W. Wellman, R. D. Krieg, "A Vectorized Elastic/Plastic Power Law Hardening Material Model Including Lüders Strain," SAND90-0153, Sandia National Laboratories, Albuquerque, New Mexico, March 1990.

J. R. Weatherby, R. D. Krieg, and C. M. Stone, "Incorporation of Surface Tension Into the Structural Finite Element Code SANCHO," SAND89-0509, Sandia National Laboratories, Albuquerque, New Mexico, March 1989.

S. W. Attaway, "A Local Isotropic/Global Orthotropic Finite Element Technique for Modeling the Crush of Wood," SAND88-1449, Sandia National Laboratories, Albuquerque, New Mexico, September 1988.

R. D. Krieg, "A Simple Constitutive Description for Soils and Crushable Foams," SC-DR-72-0883, Sandia National Laboratories, Albuquerque, New Mexico, 1978.

D. J. Bammann, "An Internal Variable Model of Viscoplasticity," *International Journal of Engineering Science*, Volume 22, pp 1041-1053, 1984.

References

¹William. C. Mills-Curran, Amy P. Gilkey, and Dennis P. Flanagan, "EXODUS: A Finite Element File Format for Pre- and Post-processing," SAND87-2977, Sandia National Laboratories, Albuquerque, New Mexico, September 1988.

²L. M. Taylor and D. P. Flanagan and W. C. Mills-Curran, "The GENESIS Finite Element Mesh File Format," SAND86-0910, Sandia National Laboratories, Albuquerque, NM, May 1986.

³*American National Standard Programming Language FORTRAN*, American National Standards Institute, Inc., ANSI X3.9-1978, New York, 1978.

⁴*American National Standard for Information Systems--Programming Language C*

References

X3.159-1989. American National Standards Institute, Inc., ANSI X3.159-1989, New York, 1989.

⁵Brian Berliner, "CVS II: Parallelizing Software Development," Paper presented at the Winter 1990 USENIX Conference, Washington, D.C., 1990.

Distribution***Distribution***

1	1400	E. J. Barsis	10	1562	G. D. Sjaardema
1	1401	J. R. Asay	1	1832	J. M. Ramage
1	1402	S. S. Dosanjh	1	2565	S. T. Montgomery
1	1403	G. S. Davidson	1	6313	J. Jung
1	1404	J. A. Ang	1	6411	A. S. Benjamin
13	1425	J. H. Biffle & staff	1	6423	J. F. Dempsey
50	1425	M. K. Smith	1	6513	D. S. Oscar
1	1431	J. M. McGlaun	1	6522	J. D. Miller
1	1431	K. G. Budge	5	7141	Technical Library
1	1431	J. S. Peery	1	7151	Technical Publications
1	1432	W. T. Brown	10	7613-2	Document Processing for DOE/OSTI
1	1433	J. W. Swegle	1	8523-2	Central Technical Files
15	1434	D. R. Martinez & staff	6	8741	G. A. Benedetti & staff
1	1500	D. J. McCloskey	1	8742	M. R. Birnbaum
1	1501	C. W. Peterson	1	8742	J. J. Dike
1	1502	P. J. Hommert	1	8742	L. I. Weingarten
1	1503	L. W. Davison	5	8743	M. L. Callabresi & staff
1	1504	D. J. McCloskey, actg			
1	1511	J. S. Rottler			
1	1511	D. K. Gartling			
1	1511	M. W. Glass			
1	1511	P. L. Hopkins			
1	1511	M. J. Martinez			
1	1511	P. A. Sackinger			
1	1511	P. R. Schunk			
1	1511	J. D. Zepper			
1	1512	A. C. Ratzel			
1	1513	R. D. Skocypec			
1	1513	R. G. Baca			
1	1513	B. L. Bainbridge			
1	1513	R. E. Hogan, Jr.			
1	1513	J. L. Moya			
1	1551	W. P. Wolfe			
1	1552	C. E. Hailey			
1	1553	W. L. Hermina			
1	1554	W. H. Rutledge			
15	1561	H. S. Morgan & staff			
13	1562	R. K. Thomas & staff			